

Shock-turbulence Interaction

Jamaludin Mohd-Yusof, Daniel Livescu, Mark R. Petersen, CCS-2

The interaction between a shock wave and an interface separating two different fluids can generate a wide variety of fluid motions. The description of the interaction combines compressible phenomena, such as shock interaction and refraction, and hydrodynamic instability, including nonlinear growth and transition to turbulence. In general, a perturbation initially present at the interface is amplified following the refraction of the shock. This class of problems is usually referred to as the Richtmyer-Meshkov (RM) instability. Accurately solving all the scales of the problem imposes severe limitations on both the spatial and temporal resolutions and, to date, no direct numerical simulation (DNS) in which all the scales of the problem are accurately resolved has been performed for the fully compressible case. However, with the current advance of supercomputers it is now possible to perform DNS of the RM instability generated by weak shocks, without the simplifications used in previous simulations. Due to various factors, including sensitivity to initial conditions, wavelength-dependent growth rates and the effects of physical and numerical dissipation (in non-DNS codes), there is considerable difficulty in understanding the differences between experimental and numerical results. Therefore the effects of molecular transport properties on the development of the instability remain outstanding open questions, which will be addressed with the current simulations.

In the first part of the project, we consider the interaction between a turbulent field and multiple passes of a shock wave for a single fluid. While this is a simpler problem than the multifluid shock-turbulence interaction, it still addresses several open questions, such as the variation of turbulence properties in the presence of an anisotropic production mechanism [1], and modeling approaches for shocked turbulence. There have been several previous studies of this simplified problem at low resolution [2,3]. Our study brings some improvements in the numerical approach and will represent the largest simulations to date of such a configuration.

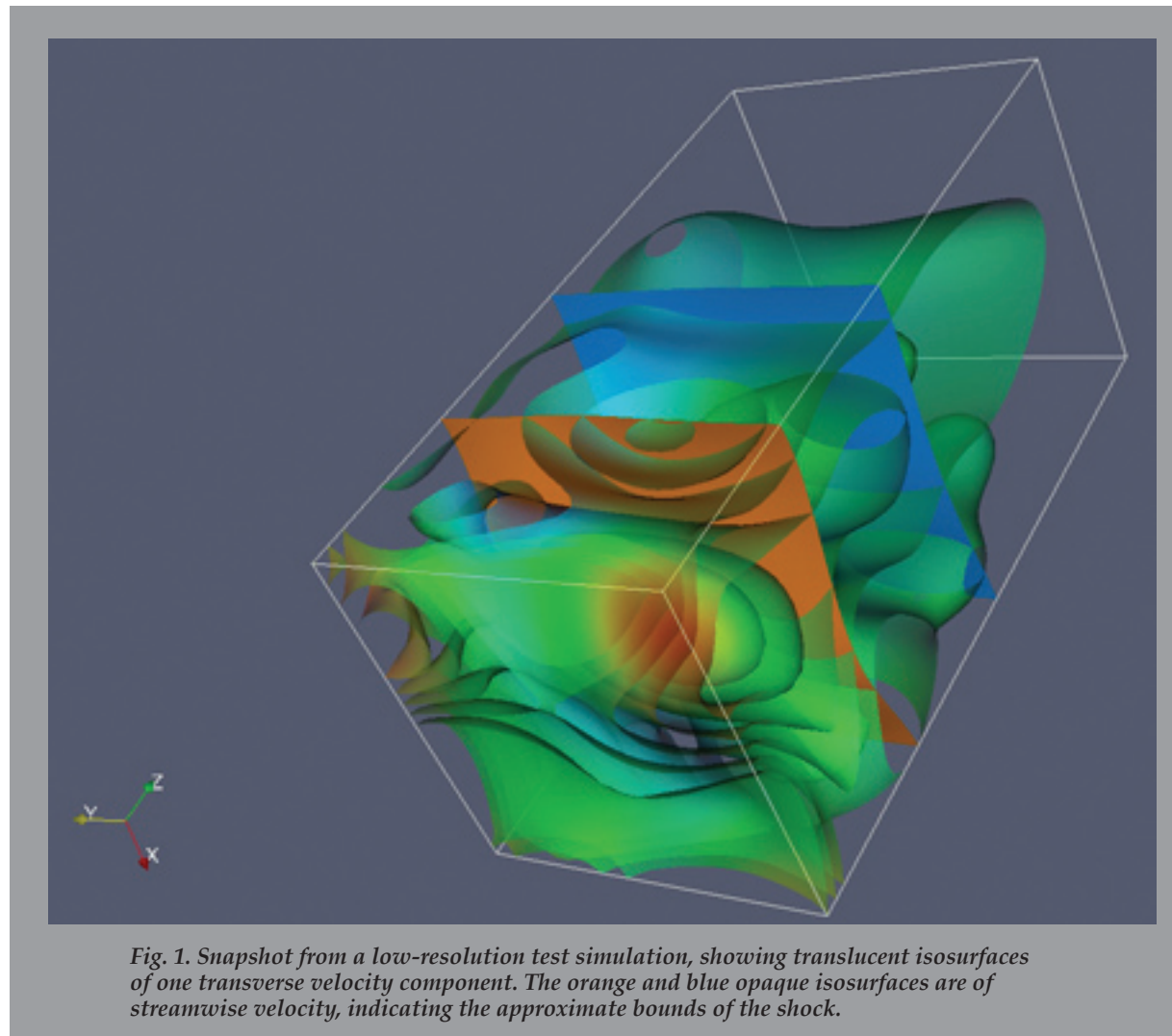
The simulations are performed using the CFDNS compressible turbulence simulation code, with periodic boundary conditions in the two spanwise directions. The inlet BC is supersonic with a turbulence velocity field generated from a tri-periodic simulation. These inlet generation simulations are performed with the

correct background velocity imposed in order to remove the ‘frozen turbulence’ approximation that would otherwise be required and was used in the previous studies [2,3]. That is, the inlet fields are snapshots of an advecting forced turbulence field at different times, rather than spatial slices of a single field at a fixed time, as done in the previous studies.

Special care is taken to ensure that spurious waves due to the outlet boundary do not affect the simulation results. We use an artificial acceleration field to convect the outgoing perturbations supersonically out of the domain, even though the flow there is actually subsonic. This minimizes the errors due to both physical and numerical waves generated at the outflow boundary.

At the time of writing, final testing of the code improvements is underway, prior to beginning the production runs. An example of the results of these tests is shown in Fig. 1.

For more information contact, Jamaludin Mohd-Yusof at jamal@lanl.gov and Daniel Livescu at livescu@lanl.gov.



- [1] D. Livescu and C.K. Madnia, *Phys. Fluids*, **1698**, 2876 (2004).
- [2] S. Lee, S.K. Lele, P. Moin, *J. Fluid Mech.* **251**, 562 (1993).
- [3] K. Mahesh, S.K. Lele, and P. Moin, *J. Fluid Mech.* **334**, 379 (1997).

Funding Acknowledgments

- Department of Energy, National Nuclear Security Administration
- Los Alamos National Laboratory Directed Research and Development Program